International Journal of Mechanical and Production Engineering Research and Development (IJMPERD)
ISSN(P): 2249-6890; ISSN(E): 2249-8001
Vol. 9, Issue 1, Feb 2019, 281-290

NON-LINEAR ELASTIC-PLASTIC BEHAVIOUR OF ALUMINIUM SHEET METAL USING FINITE ELEMENT ANALYSIS

HAKIM S. SULTAN ALJIBORI

College of Engineering, University of Warith Alanbiyaa, Karbalaa, Iraq

ABSTRACT

Aluminium is one of the most used metals in today's industry, having properties of strength, durability, conductivity, lightness, and corrosion resistance. Sheet metal forming is a process that widely used and costly manufacturing process. A materials problem is one of selecting the right material from the many thousands that are available. Aluminium has a low density, good corrosion resistance and relatively cheap. Aluminium sheet becomes favourable comparing with steel regards to some improvement at the designs. Automotive parts and products are used wide range of these materials included bumpers, doors, bars, seat frames and roof panels. Nonlinear analysis is much more complicated than simple linear analysis because it is required many variables such as changes in geometry, permanent deformations, structural cracks and buckling. This paper was carried out to study the elastic-plastic analysis of sheet metal forming using finite element method. LUSAS simulation was carried out to understanding the behaviour of aluminium sheet and accurate results of this process. Axi-symmetry and plain strain element mesh were used to model and study this metal. Deep analysis was carried out and the effect of geometry of sheet metal forming process has been studied. A good agreement between the load and the displacement test was obtained that verified the program.

KEYWORDS: Elastic-Plastic, FEA, Aluminium, Geometry & LUSAS Simulation

Received: Oct 29, 2018; **Accepted:** Nov 19, 2018; **Published:** Jan 07, 2019; **Paper Id.:** IJMPERDFEB201927

Original Article

1. INTRODUCTION

The sheet is plastically deformed by stretching, bending and drawing in the forming process like, press.

The computer tools allow the design engineer to investigate the process and material parameters controlling the material flow over the die surfaces. The most of automotive parts are still manufactured by the forming technology of the sheet metals [1]. Taylor L and et al., explain the main parameters of sheet-metal forming by numerical simulation, with the increasing of finite element (FE) and simulations in automotive companies, the analyses of sheet metals are performed in the design feasibility studies of production tooling and stamping dies [2]

There is a limiting factor in product design of producing the desired shape in a forming process, for example, with no cracks or wrinkles in the sheet metal. In aerospace and automotive industries are growing rapidly for precise and accurate parameters concerning parts design, so the metal sheet becomes essential for this process. Simulation of sheet metal forming is plays an important role in manufacturing processes. In parallel with concurrent engineering, forming process of sheet metal is proving to be the main tool in linking design and manufacturing. Under some conditions, such plane strain sheet metals were to be failing under this condition, several plane strain FE (finite element) codes for sheet metal forming analysis has been studied by M. J. Saran, Frey and Wenner, M. P. Sklad [2-4]. in the absence of shape tooling curvatures, the codes are capable of predicting deformation very accurately and very fast, but as tooling curvature cause significant bending strains, these codes

can no longer capable to predict accurately [5]. S.K. Panthi et al. studied the effect of load on spring-back of total-Elastic-Incremental-Plastic Strain of sheet metal bending process using FE and experimental analysis, the prediction of spring-back and the results are presented in terms of spring-back ratio, The numerical results agree with the experimental ones with a reasonable accuracy [6].

Math and Grizelj [7] examined the spring-back and residual stresses of steel plates, this plate designed for spherical tanks that made of steel, using elastic–plastic behaviour using FE and experimental validation. Another model of plasticity follows kinematic rules defined by a back stress evaluation equation together with a hardening modulus function. Prager [8, 9] proposed a kinematic hardening rule postulating that back stress with an increment of plastic strain through a single material constant and yield surface moves in the direction of plastic strain increment in the stress space. The yield surface translation is linearly related to the plastic increment, this hardening rule is called as ‘‘linear kinematic hardening

The main objects of this study focused on Finite element simulating of non-linear sheet forming process in order to understanding of metal sheet flow in the manufacturing forming process.

2. BENDING AND SPRING-BACK PROCESS

Generally, the Schematic Diagram of Punch and Die is shown in Figure 1. The force required to punch is mainly the shear strength of the sheet and the area being sheared. The length of the axis is used to determine the blank length for the bent part. The approximate formula for the bend allowance (L_b) is given by

$$L_b = \alpha(R + Kt) \tag{1}$$

The above symbols, (α) is the bend angle in radians, (R) is the bend radius, (K) is a constant factor (K=0.33 for R< 2t and K=0.5 for R> 2t) and (t) is the thickness of sheet. The minimum bend radius is the radius at which the crack appears on the outer surface of the bend. In the elastic limit, the outside of the bend is in tension case and the inside of the bend is in compression case. The neutral axis half way through the sheet thickness is remaining (Figure 2). At bending process becomes plastic, the position of axis changes and then moves towards the concave side of the bend [10], thus the tensile strain is more than or greater than the compressive strain. When the radius R decreases, then strain (ε_1) and strain ($-\varepsilon_3$) increase and thinning become more obvious until finally cracking can occurs.

$$\varepsilon_1 = -\varepsilon_3 = \ln\left(1 + \frac{1}{\frac{2R}{t} + 1}\right) \tag{2}$$

Spring-back in bending is shown in Figure 3, Hosfold and Caddeell [11] have propose for the simple bending case that

$$\frac{1}{R_0} - \frac{1}{R_f} = \frac{3\sigma_{ps}}{tE} \tag{3}$$

Where, R_0 is the original radius of curvature, R_f is the radius of curvature after spring-back, $\boldsymbol{\sigma_{ps}}$ is the yield stress, t is the thickness of plate and E is the elastic modulus.

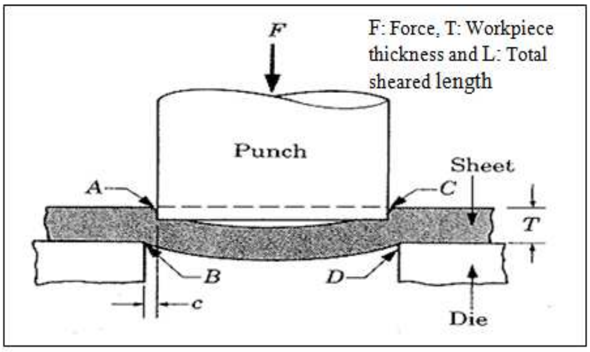


Figure 1: Schematic Diagram of Shearing With Punch and Die

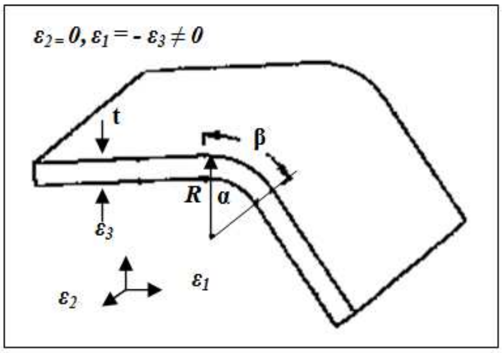


Figure 2: Bending Processing

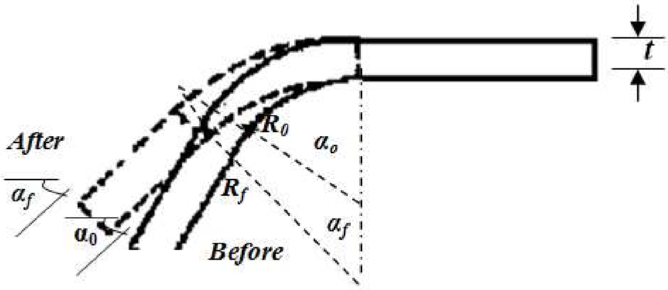
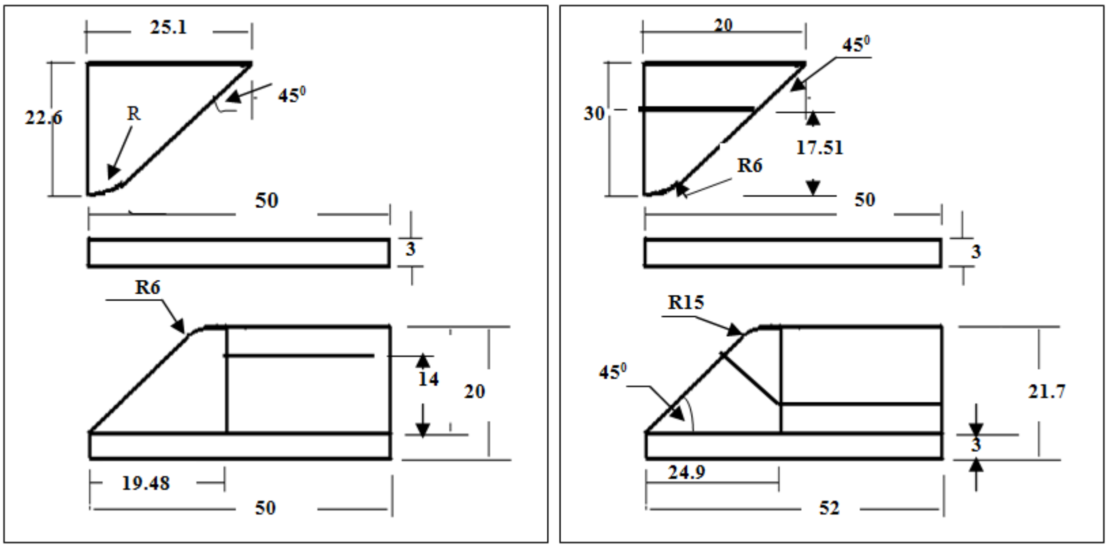


Figure 3: Spring-back in Bending



a) V-Die Bending with Die Radius R6 **b) V-Die Bending with Die Radius R15**

Figure 4: Schematics Diagram and Dimensions of the Models

There are complex interactions between the specimen (such, tolerance, geometry, surface, etc.), the forming process (forming, tooling, force, machine, lubrication, and so on) and the material (material parameters, ductility, residual stress, structure, corrosion resistance, etc.) which exist in forming processes. So, the studies of these processes were interesting by many researchers [12], a larger radius of the die causes spring-back. Therefore, in the first stage of tool design used a smaller radius 6mm as the initial condition for the analysis. The punch stroke suggested by Meguid and Refaat [13] is 35mm. But eventually throughout the modelling exercise it is proved that 35 mm is not suitable for the model in use and some modification has been applied on the model itself, including the punch stroke, length between symmetry line to the centre of die curvature and draft angle of the die generally (1^0 □□2^0*)* degrees. At the view point of

economic consideration the thickness of the die should be optimize to withstand the applied and cyclic load. The standard sheet thickness 3mm is taken as default throughout the computational experiment. After considering the final dimension of the die geometry was produced using Computer Aided Design (CAD) software, the drawing is in 2D since the analysis is carried out in 2D simulation as well. Since there is some inaccuracy while exporting the geometry of the model therefore it was decided to transfer the drawing manually using the coordinate (with the accuracy of 4 decimal points) of the nodes. Figure 4 shows Schematics diagram and dimensions of the models.

3. FINITE ELEMENT SIMULATION

The finite element method (FEM) is a numerical method for solving problems of engineering and mathematical physics. Finite element method is a simulation technique which evaluates the behaviour of structures and components for different loading conditions and material properties including applied loads, temperatures and pressures. FEA used for predicting how a real object will react to forces, vibration, heat and so on. The analysis of the component or any structure is obtained by simultaneous analysis the individual finite elements, due to their individual positions within the mesh and being totally dependent upon the assistance of an automatic computer [14]. Numerical modelling of metal forming processes became possible to simulate the metal deformation and to calculate stress and strain and deformation states for complex processes. By using this method, simulation of design can help us also to predict errors and then modification can be done at an early stage before the parts was fabricated and tested. In finite element methods the labour cost and time lost can be reduced to minimum limits. Therefore, finite element methods will gradually replace manual trial-and-error design iteration with numerical simulations in future. It can be seen that the increasing demands of using finite element method in the manufacturing process (especially in pre-processing analysis stage) will greatly enhance the efficiency and saving of time and manpower.

In this simulation and model development complete LUSAS Finite Element software was used in in this test that consists of three main stages are:

- Pre-Processing stage
- Solver (Finite Element) stage
- Post- processing (Results-Processing)

LUSAS software is a general purposes finite element analysis system which is consist of ma several analysis facilities, can be use it to solve variety of linear and non-linear behaviour, dynamics and thermal engineering analysis problem and also can be used for composite material [15]. LUSAS software is capable to model and analysis the structural problems by assembling all of the simple expression into a set of simultaneous.

Sheet metal forming processes are the complex interaction between specimen (geometry, tolerances, surface topology, etc.), the forming process (tooling, forces, lubrication, etc.) and the material (ductility, material parameters, micro structure, corrosion resistance, residual stresses etc.), which exist in forming processes. There are many changes areas and the reason why the interactions in sheet metal forming processes are so complex is that a change in one area creates changes in the other areas and that the interactions are non-linear behaviour. Initially, the sheet is used uniformly regular finite element mesh. Along the forming process, incremental displacement was applied throughout the iteration steps was carried out. Then the parts of the mesh are refined as necessary. The contact between the points of surface is very

important in FE methods. So, to determine the contact points between the forming tools and the sheet metal in order to ensure smooth flows of material, a proper consideration has been carried out regards to punch and die radius. Additionally, large plastic deformation has been modelled using material laws with the correspond strain hardening. Since the model is symmetry therefore; only half of the model is used and generated to simulate with time saving. The mesh generated and boundary conditions of the model were shown in Figures 5 and 6.

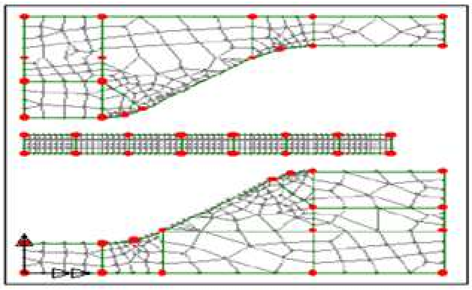


Figure 5: Geometry and Mesh Generation of Model

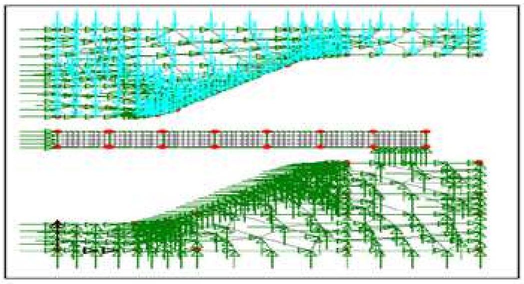


Figure 6: Load and Boundary Conditions of Model

3.1 Contact Procedures

LUSAS Finite Element software has the capability to solve non-linear problems, there are three parts of this analysis should be brought into contact before the running analysis, these parts are; the punch, sheet metal and die. Starting with two points from the sheet and the edge of the die to move it in the contact together, Then the same method can be applied to move two points from the punch with different sets of these points as shown in Figure 7. In the contact procedure only the expected point, surface or region of contact should be selected as a slide-line analyses and a suitable meshing in the region of contact should be selected. Sharp shapes or corners are described as two slides. The final model can be shown in Figure 8. This Figure should be ready for running the program and starting with the analysis. After that finite element analysis of the body or selected part will stare with the simulation for some time until completion of analysis. If there are any errors in the analysis, the program will stop and modification or correction should be made to start the analysis again. Further moving down of the punch; the free end of metal sheet will exposed to some compression. The penetration may be occurring at a very small scale and it is still occurs also with some lubrication when the punch or die are making contact with the metal sheet. The final model of geometry, mesh and deformation of metal sheet can be shown in Figure 9.

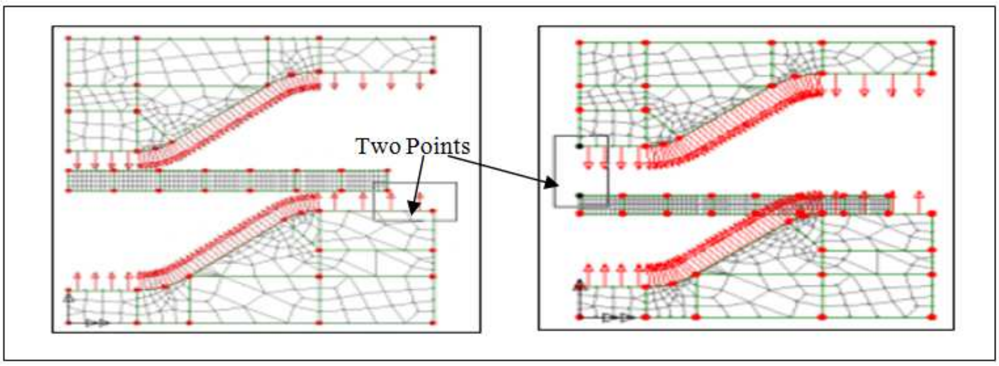
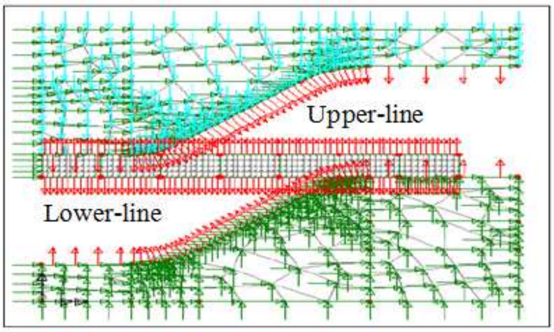


Figure 7: Two Points Selected to Invoke Precise Translation



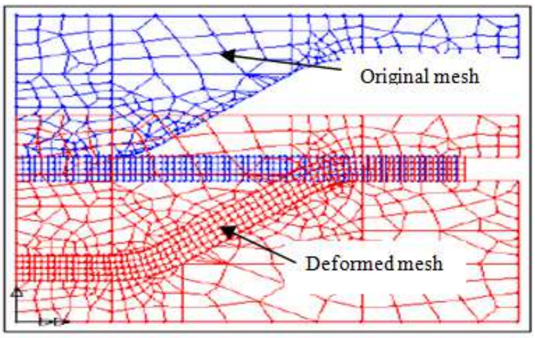


Figure 8: Final Model of Analysis Figure 9: Original and Deformed Mesh

3.2 Material Properties

The statement that in the elastic region, stress is independent of material properties, and that it becomes material-dependent when plastic deformation begins to occur. In this section will refer to material properties that used to the current study; there are two kinds of material (isotropic material) are: linear elastic (LE) for the punch and die and linear elastic-plastic (LEP) for sheet metal. The basic material properties that used and it were required for this analysis are tabulated in Table 1

Table 1: Material Properties of the

Material Property	Symbol	Property Value
Sheet Metal (Aluminium) Linear Elastic-Plastic		
Young's Modulus	E	78 GPa
Uniaxial Yield strength	σ_y	550MPa
Poison's Ratio	ν	0.3
Material Density	ρ	2700 kg/m^3
Punch and Die (Steel), Linear Elastic		
Young's Modulus	E	400 GPa
Uniaxial Yield Strength	σ_y	4000MPa
Poison's Ratio	ν	0.3
Material Density	ρ	7800 m^3

3.3 Nonlinear Analysis

As a linear finite element analysis (simple linear) means all materials are assumed linear elastic (LE) behaviour and the deformations and strain are very small effect, which is mean there are no significant effects of overall behaviour of the model. While; nonlinear finite element (NLE) analysis is sophisticated and more complex than simple linear analysis due to being many variables and directions such as changes in geometry, material cracking, buckling and type of deformations. The main difference between linear analysis and nonlinear analysis of geometry is that in equilibrium linear analysis is the initial configuration un-deformed model. While in nonlinear analysis should be satisfied in the deformed configuration [9]. In this paper and using a finite element program (LUSAS finite element software) with two types of nonlinear analysis are: *Geometric nonlinearity* and *material nonlinearity.* In Geometric nonlinearity, in which emerge from great changes in the structural configuration during loading and material nonlinearity in which effects emerge from a nonlinear constituent model that is, progressively dis-proportionate stresses and strains). Nonlinear material and geometric effects may be combined with the current analysis. To obtain the equilibrium state in a nonlinear analysis, the model can solve it many times, and according to equilibrium state; adjusting the applied load based on the current state and modification in the geometry can make it according to displacements. Step-by-step convergence procedure of nonlinear

analyses and it is generally solved by mathematical procedure (iterative procedure) that is uses an initial guess to generate a sequence of improving approximate solutions and the level of convergence reached tells us the error in the solution. Some of nonlinear analysis is not guaranteed to converge; but most of convergence may be helped along by modifying some of the factors or parameters as well as in nonlinear analysis, it is not always possible to obtain the correct results according to external loads. This analysis allows you to put any load levels for each increment or time step of a nonlinear solution.

4. RESUTS AND DISCUSSION

The results obtained from LUSAS Finite Element analysis of deep drawing sheet metal forming process to analyse the prescribed model to obtain the output files. This analysis was carried out with a series of parameters and the effect of geometry sheet metal forming process has been identified and studied. V-die type bending was performed and the influence of die radius with regard to force applied was examined and investigated. Many simulation tests were implemented to obtain the most acceptable value for some of the parameters. From the results one can show that the Model 2 have greatest punch stroke if comparing with the Model 1, therefore; the resulting in stress was higher. Noticed that the stress also at free end of the axi-symmetry model but intensified at curve region. The stress distribution for plane strain element is steadily generated over the area between two-curve regions. Regarding model 1 does not effect from the thinning influence and sliding-sticking condition that occurs on the Model 2. Unusual or infrequent formation of the final dimensions due to the penetration that occurring. For getting greater displacement or stroke should be there is high load is required. Equivalent Stress -SE- contours for models, 1 and 2 as shown in the most of Figures (from Figure 10 to Figure 17) for plain element mesh and axi-symmetry element mesh. As well as from the contours shown in plotted results one can notice that there are higher stress provided from displacement test comparing with loading test, since the force was directly applied to the punch. A high applied load is required for plain strain model to causes the metal sheet to achieved the hole or die cavity. The stress concentrate at the middle portion of the Aluminium metal sheet for plain strain model that is totally different from the high stress concentration exist at the curvature bending region of metal sheet for the axi-symmetry model. At the same way, the results of the V-die models can be obtained from similar analysis. The additional dimension of die radius causing decrement in the load and increasing the maximum stress and the Figures 18 and 19 shows the results of stresses of the deformed shape and load-displacement curve was shown in Figure 20 with the effect of die radius in V-die model.

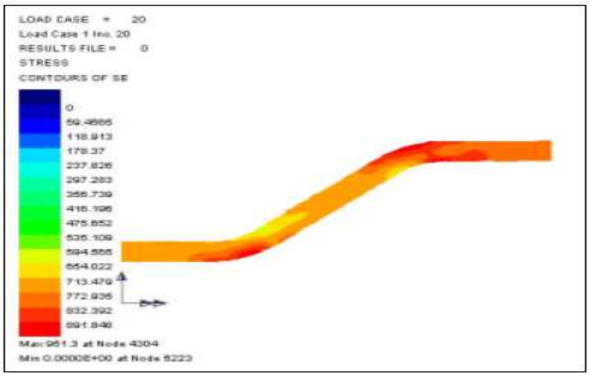


Figure 10: Axi-symmetry Model Displacement Test for Model 1

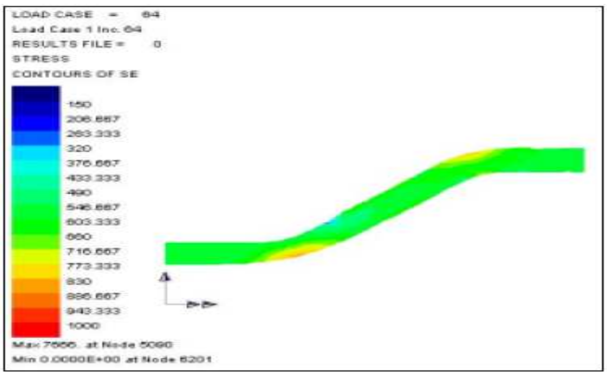


Figure 11: Axi-symmetry Model Loading est for Model 1

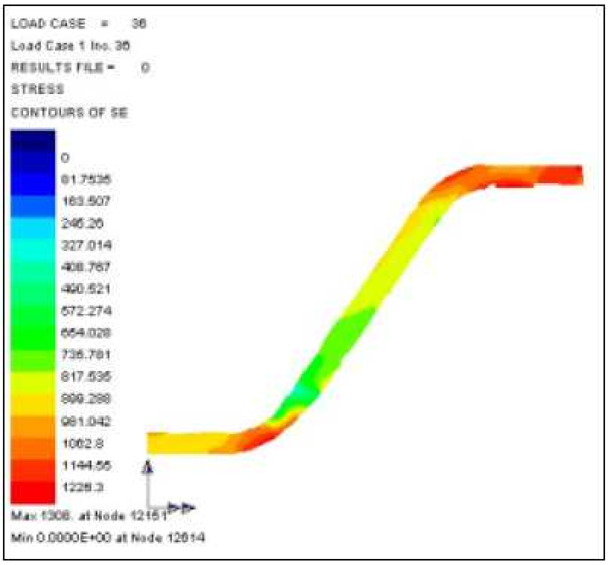


Figure 12: Axisymmetry Model Displacement Test for Model 2

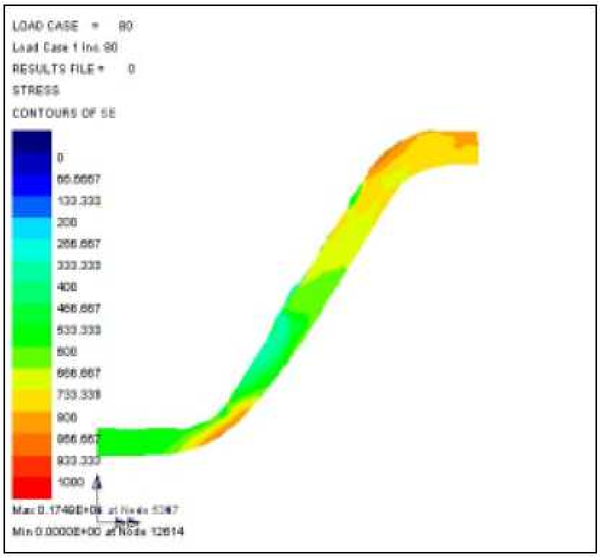


Figure 13: Axisymmetry Model Loading Test for Model 2

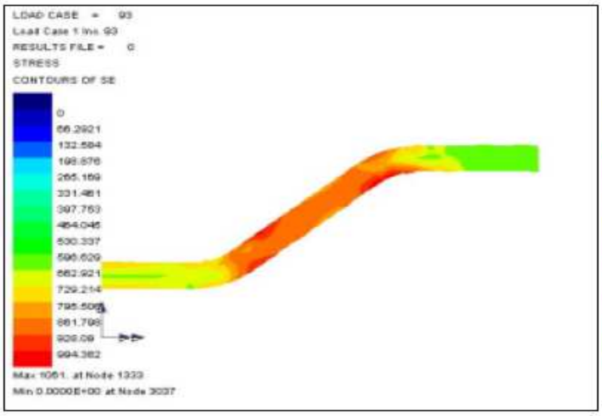


Figure 14: Plain Strain Model Displacement Test for Model 1

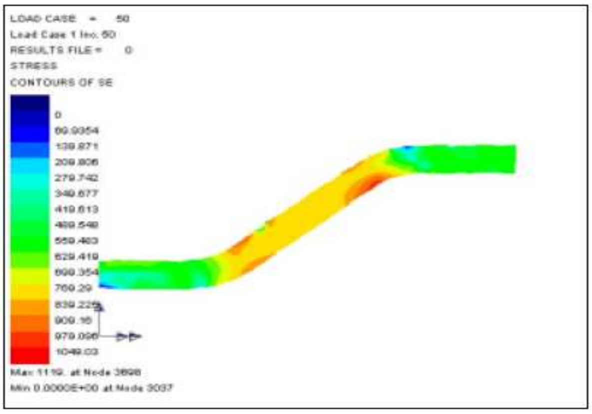


Figure 15: Plain strain Model Loading Test for Model 1

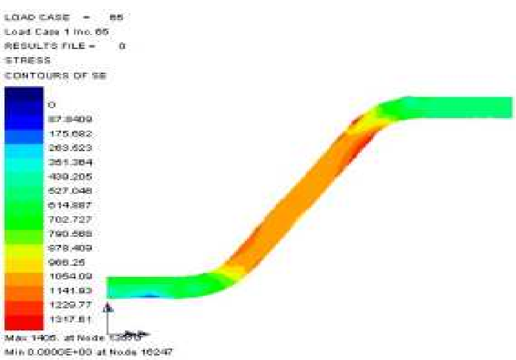


Figure 16: Plain Strain Model Displacement Test for Model 2

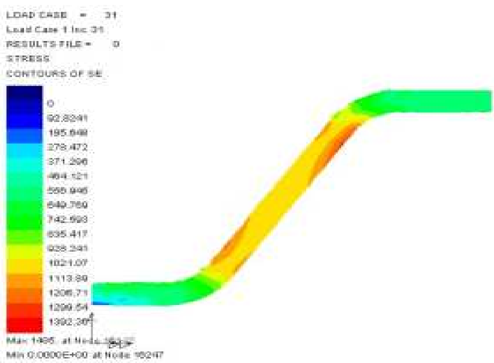


Figure 17: Plain Strain Model Loading Test for Model 2

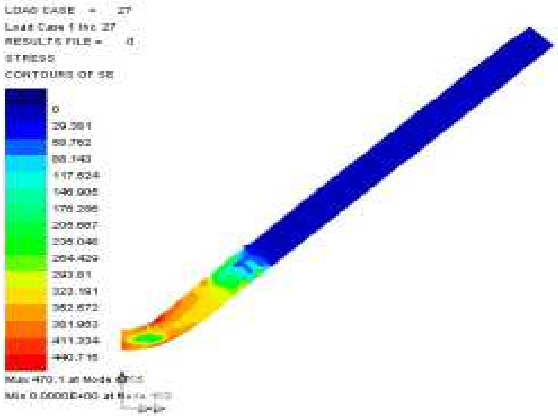


Figure 18: Plane Strain V-Die Model (Rd 6) for Loading Test

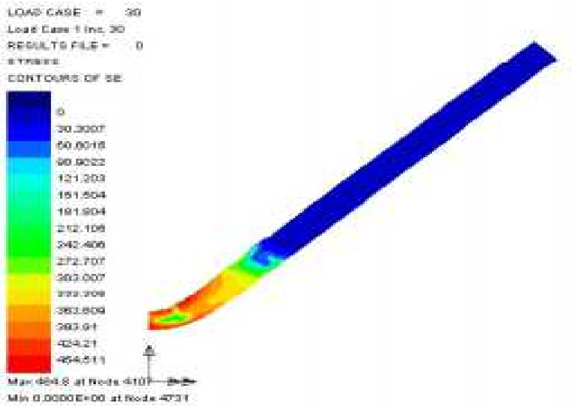


Figure 19: Plane Strain V-Die Model (Rd15) for Loading Test

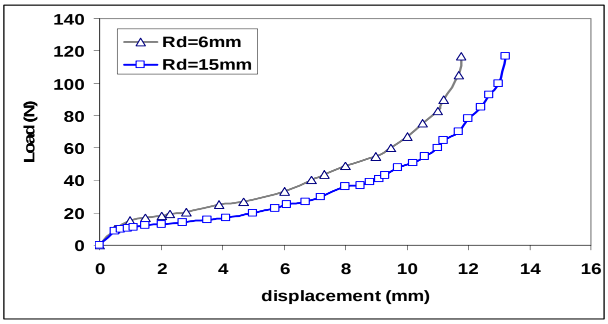


Figure 20: Load-Displacement Curves and Effect of Die Radius for V-Die

5. CONCLUSIONS

By using finite-element analysis and simulation in the manufacturing process and forming operations is becoming more important and businesslike as it provides a successful and cheap way to determine and analysis most of the important parameters. This work presented and identified a finite element simulation study with regard to the aluminium sheet metal forming. Deep drawing analysis was modelled and take place the effect of geometry towards the sheet metal forming process. V-die bending was carrying out and the effect of die radius towards force applied was examined. Many FE tests were carried out to obtain the most suitable values for some of the effective parameters. The effect of process parameters on stress distribution and punch load were tested and elastic-plastic finite element computational software (LUSAS Software) was developed to simulate the deep drawing process and V-die bending sheet metal process. Satisfactory acceptable and agreement between the loading test and the displacement test was obtained that verified and confirmed the program. Some conclusions of the current study can be summarized as follows:

- The slide - line option was used by element mesh for simulation as a constraint.
- At the bending region, higher punch stroke was induced and the stress distribution appeared along the metal sheet.
- Sliding-sticking condition becomes visible as the forming process induces higher changes of sheet dimensions. Plasticity extension and elongation occurs in the material; especially in the contact line of bending region.
- Radii of Punch and die designed in order to maintain and ensure a smooth flow of material to and good quality of the final product.

- Excessive displacement and too much load of punch shall be avoided ensuring that the sheet is not penetrating the punch or die.
- When the die radius increases the punch load decreases for V-die.

6. REFERENCES

1. *M. Firat, U-channel forming analysis with an emphasis on spring back deformation, journal of Materials and Design 28 (2007) 147–154*
2. *M. J. Saran and R. Wagoner. (1991), A user's manual and verification examples, sheet stamping. Report no. ERC/NSM-S-91-17*
3. *W. H. Frey (1987). Development and application for a one dimensional finite element code for sheet metal forming. Materials Processing, V.1 pp.307-320, Boston*
4. *M. P. Sklad, (1990), 16th IDDRG Congress, ASM International, Sweden. pp. 295*
5. *F. Pourboghrat (1992) Spring-back calculation for plain strain sheet forming using finite element membrane solution, ASME 1556*
6. *S.K. Panthi, N. Ramakrishnan, K.K. Pathak, J.S. Chouhan, An analysis of springback in sheet metal bending using finite element method (FEM), Journal of Materials Processing Technology 186 (2007) 120–124*
7. *Yunus, M. O. H. A. M. M. E. D., & Alsoufi, M. S. (2015). A statistical analysis of joint strength of dissimilar aluminium alloys formed by friction stir welding using taguchi design approach, anova for the optimization of process parameters. IMPACT: Int J Res Eng Tech (IMPACT: IJRET), 3(7), 63-70.*
8. *M. Math, B. Grizelj, Finite element approach in the plate bending process, J. Mater. Process. Technol. 125–126 (2001) 778–784.*
9. *Prager W. A new method of analyzing stresses and strains in work-hardening plastic solids. J Appl Mech 1956; 23:493–6.*
10. *Nagaral, M., Auradi, V., & Ravishankar, M. K. (2013). mechanical behaviour of aluminium 6061 alloy reinforced with al2o3 & graphite particulate hybrid metal matrix composites. International Journal of Research in Engineering & Technology (IJRET) Vol, 1, 193-198.*
11. *Ziegler H. A modification of Prager_s hardening rule. Quart Appl, Math 1959;17:55–65*
12. *Roger Pearce (General, International Deep Drawing Research Group) Sheet Metal Forming. New York, pp81-88*
13. *Niyas Salim, M. A., & Kumar, A. (2014). Effect of Mg Enhancement in the Microstructure and Mechanical Properties of AC2A Aluminium Alloy. International Journal of Research in Engineering & Technology, 2, 139-148.*
14. *Hosford W F and Caddell R M (1998), Metal Forming (Englewood Cliffs NJ: Prentice Hall)*
15. *You-Ming Huang and Daw-Kwei, Effects of process variables on V-die bending process of steel sheet. Int. J. Mech. Sci. Vol.40 No. 7, pp 631-650,*
16. *S.A. Meguid and M.H. Refaat (1997), Finite element analysis of the deep drawing process using variation inequalities Elsevier Science*
17. *Hakim S. Sultan, (2004), Experimental Investigation and Finite Element Analysis of Composite Conical Structures" PhD thesis, University Putra Malaysia, Malaysia*
18. *LUSAS/standard User's Manual, LUSAS theory manual, version 13.5. FEA United Kingdom*